Fusion3

ALL FUSION3 PRINTERS

GETTING STARTED WITH F3SLIC3R

Revision 2/13/2026

INSTALLING, CONFIGURING, & USING F3SLIC3R

This short guide will get you started with the basics of how to install and use Fusion3's slicer, F3Slic3r.

F3Slic3r is based on Prusaslicer by Prusa Research, which in turn is based on the open-source project slic3r and other open source contributions. F3Slic3r and Prusaslicer are part of a family of slicers with the same common heritage, so if you have used other 3D printer slicers you should be at home fairly quickly.

DOWNLOAD

Latest Version (recommended)

All printers can use this version.

Download 2.8.2:
https://www.fusion3design.com/wp-content/uploads/2025/09/F3Slic3r-setup-2.8.2.0.zip

If you require a Mac version, please see the section on alternate slicers

Old Version

EDGE, F410, and F400 printers can use this version. **F200 CANNOT USE THIS VERSION.** If you cannot get the latest version to work, please try this.

Download 2.6.x:
https://www.fusion3design.com/wp-content/uploads/2023/07/F3Slic3r-1.0.10.zip

INSTALLATION

See also: https://www.youtube.com/watch?v=w3l7Vgb0iKc

Clean Installation (PCs only)

Use these steps if you have never installed F3Slic3r before.

1. Run the installer. You should not need to do anything special.

2. After installation is complete run the application and go to the "First run" section of this document

Update Installation (PCs only)

Use these steps if you have installed F3Slic3r before and are updating to the latest version.

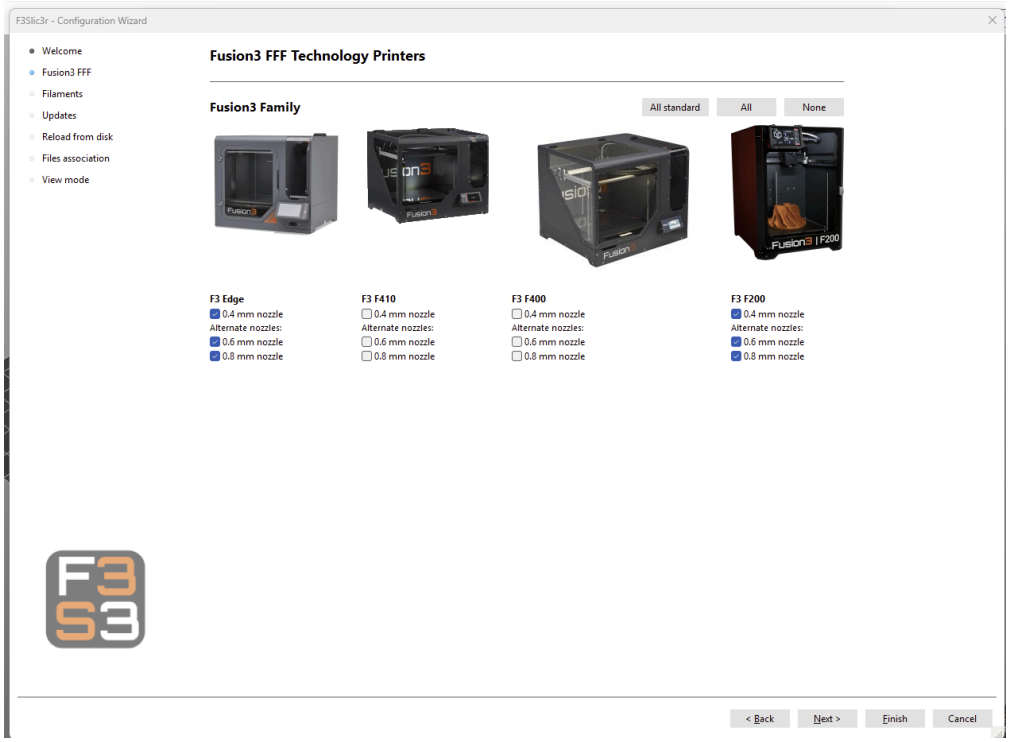
1. Make a backup of any custom profiles you have created
    a. Start F3Slic3r
    b. Go to ***Help > Show configuration folder***
    c. Copy any profiles you want to keep out of these 3 folders: ***filament, print, printer***. You can ignore the other folders.
    d. Put them someplace you can find them on your hard drive.
2. Uninstall your current version
    a. Control Panel > Programs and Features
    b. Locate **F3Slic3r** in the program list and uninstall it
3. Clean up remaining system files
    a. Open windows explorer. Type "**%appdata%**" in the location bar and press ENTER.
    b. Locate the folder called "**F3slic3r**". Delete it.
4. Install the new version as if it's a clean install
5. Reload your custom profiles
    a. Do this after starting the new version of F3Slic3r for the first time and completing the Configuration Wizard.
    b. Go to ***Help > Show configuration folder***
    c. Paste your custom profiles back into their respective folders (filament, print, printer)
    d. Restart F3Slic3r and confirm they show up in the profile list.

FIRST RUN

See: https://www.youtube.com/watch?v=w3l7Vgb0iKc

The first time you run the application the configuration wizard will run. Use this to configure the software for the printers you have.

Your configuration wizard should show 4 printers like this:



DIRECT CONNECTION TO PRINTER

You can directly connect your printer to your slicer in some cases. See: https://youtu.be/d6CYlfcCGdk

(this is currently broken in v2.8.2 because of an issue with the installer. It will be fixed in a future version)

BASIC USE

See: https://www.youtube.com/watch?v=w3l7Vgb0iKc

Picking the Right Profiles

It's vitally important that you select the right profiles in F3Slic3r:

- "Printer" must match the physical printer you're using.
- "Filament" must match the material you have loaded into the printer.

The printer DOES NOT SAFETY CHECK what material you have selected in your gcode vs what's loaded in the printer.

We have created handy lookup tables to help you with this.

- For EDGE, F410, F400: CLICK HERE
- For F200: CLICK HERE

CREATING OR MODIFYING PROFILES

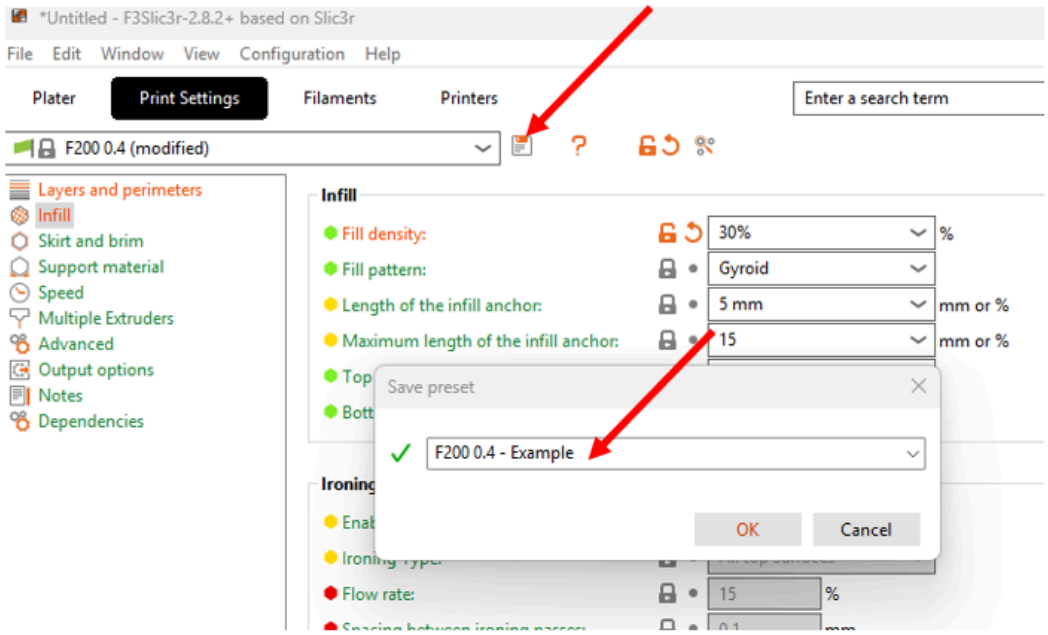
Our included "manufacturer" profiles will not let you save changes to them. This is for consistency - you will always have a known-good profile to refer back to.

If you want to save changes to these profiles to use them repeatedly, you'll need to dive into something called "profile dependencies" in order to use them with your printer. Here's the step-by-step process.

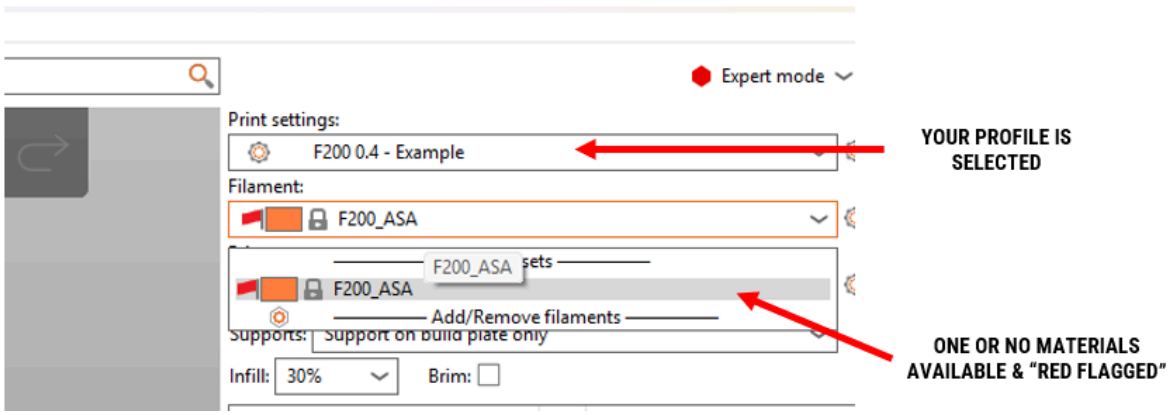
Modifying Print Settings Profiles

1. On the ***Print Settings*** tab, make the changes you want to keep in your own profile. In our example we're going to change the infill and layer height.

2. Click the SAVE icon next to the profile name. Enter a new name.



3. If you go back to the ***Plater*** view, you'll notice your new profile is selected in the "Print Settings" drop-down. BUT, none of the materials for your printer show up. This is because we have to update the dependencies.

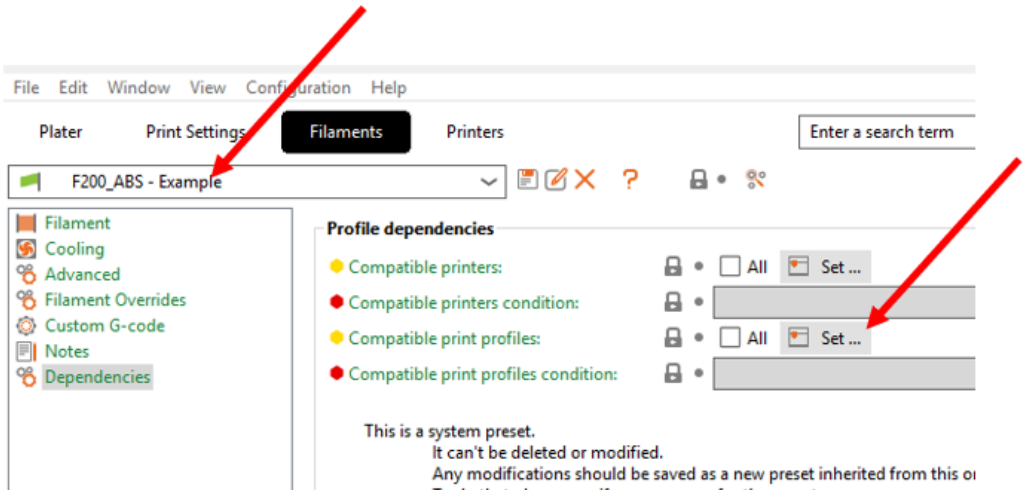


4. Select the manufacturer preset for your printer and nozzle size again.

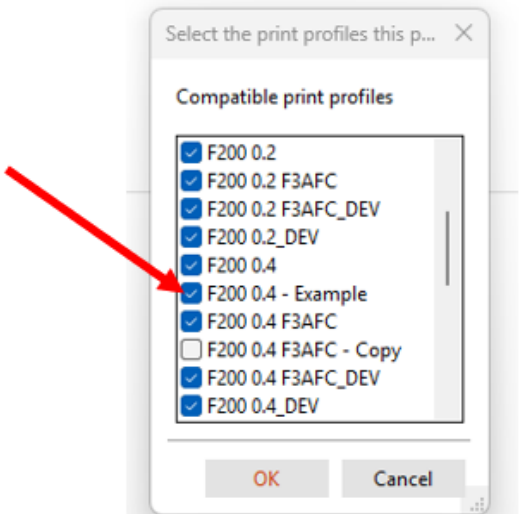
5. Then select the material you want to print with. In our example we'll use ABS.

6. Go to the ***Filaments*** tab and create a copy of the material. Do this even if you are making no changes to any of the filament settings.

7. Go to the ***Filament > Dependencies*** tab. On the "compatible print profiles" line, click the "Set" button.



8. In the print profiles list, find and check the custom profile you just made. Click OK. Save the Filament profile.



9. Go back to the Plater tab and confirm that:
    - Your custom Print Settings profile is loaded
    - Your desired material is loaded
    - No red flags on any profile list

Modifying Material Settings

Repeat steps 5-9 above with any changes to the ***Filament*** settings you want to make.

ALTERNATE SLICERS

In some cases you will not be able to use F3Slic3r for certain reasons. Maybe you need a Mac build, or maybe your company's security team needs to review every new piece of software. In these cases, it may be appropriate to use an alternate slicer.

Prusaslicer

F3Slic3r is heavily based on Prusaslicer, so in many cases it's possible to use Prusaslicer without feeling like you need to learn a new piece of software.

To use Prusaslicer with Fusion3 printers, you will need to side-load our profiles into the software (covered below).

We recommend using the same version as F3Slic3r if possible, to minimize the chance of the profiles not working correctly.

Download Prusaslicer:
https://prusaslicer.net/#download-prusa-slicer

Mac Prusaslicer Installation

1. Download the MacOS install file and run the DMG installer by double clicking it, then dragging the icon to the Applications folder.
2. The program will show as “Prusa Original Drivers” in the Applications menu. Expand that to show the PrusaSlicer program itself.
3. Once the program opens, cancel out of the configuration wizard and go to ***Help > Show configuration folder***
4. Use the below steps to add the filament, printer and print profiles into PrusaSlicer. The best way to transfer the files and by opening a new finder window in addition to the configuration folder window, then drag the files between the two windows.

Side Loading Profiles (PCs & Macs)

1. Download our profiles here (all printers): CLICK HERE

2. Unzip this on your computer somewhere you can find it.
3. Open Prusaslicer. Cancel out of the configuration wizard if it tries to run.
4. Go to ***Help > Show configuration folder***
5. Copy the profiles you need for our printers out of the unzipped folders into the Prusaslicer profile folders (filament, print, printer).
6. Close and restart Prusaslicer and confirm you see the profiles in the list.

Note: These profiles will have "_DEV" in the name at the end, but they work the same as our system profiles.

GETTING HELP

To get help or report issues or bugs, please reach out to our support team:

support@fusion3design.com

https://www.fusion3design.com/contact-support/

877-452-0010 x2